

**LARGE EDDY SIMULATION  
OF THE SOUND EMISSION  
FROM UNSTEADY FLOW SEPARATION**

NAG-1 1880

*GRANT/TR/IN/71*

Final Report

Ugo Piomelli  
Department of Mechanical Engineering  
College Park, MD 20742

## 1 Project summary

This research aimed at applying calculating the sound emission from a driven cavity using LES.

## 2 Accomplished tasks

During the course of this research, first we have determined the accuracy of the available subgrid-scale models in non-equilibrium flows. It was found that the Lagrangian eddy-viscosity model and the Lagrangian mixed models give the most accurate results. These results were published in the two enclosed articles [2, 1].

Secondly, we attempted to use a spectral multi-domain code for the problem under consideration. The effort was, however, unsuccessful. The code developed instabilities at the sharp corners of the cavity due to discontinuities in the velocity derivative.

Then, a finite-difference, staggered code was developed. The code was tested by computing the flow-field in the flat-plate boundary layer studied numerically and experimentally by Spalart and Watmuff [3]. The immersed-boundary technique [4, 5] has been implemented and tested both in laminar and turbulent cases. This technique consists in applying distributed body forces inside the computational domain. These forces are designed in such a way that the fluid velocity goes to zero (or to some assigned value) at specified points.

The “Direct forcing” method [5] has been used. This method consists in the following steps: within the fractional time-step framework, the velocity is solved from the Helmholtz equation:

$$\frac{v_i - u_i^n}{\Delta t} = \text{RHS}_i + f_i.$$

The body force is then assigned to be zero everywhere except on the immersed-body surface,  $\Gamma$ , where it is given by

$$f_i(\mathbf{x}_s) = -\text{RHS}_i + \frac{V_i - u_i}{\Delta t},$$

where  $V_i$  is the body velocity. This forces the no-slip condition to hold on the predicted velocity at the internal points. The body force was implemented via a linear interpolation of the velocity between the body itself and the second grid-point outside the body[5].

With this method, the pressure gradient on the immersed body itself is zero; this can be shown very easily by considering the Navier-Stokes equations in a  $\tau - n$  frame of reference that moves with the body, where ( $\tau$  is tangent,  $n$  normal to the body surface):

$$\begin{aligned} \frac{\partial u_n}{\partial t} + u_\tau \frac{\partial u_n}{\partial \tau} + u_n \frac{\partial u_n}{\partial n} &= -\frac{\partial p}{\partial n} + \nu \left( \frac{\partial^2 u_n}{\partial \tau^2} + \frac{\partial^2 u_n}{\partial n^2} \right) \\ \frac{\partial u_\tau}{\partial t} + u_\tau \frac{\partial u_\tau}{\partial \tau} + u_n \frac{\partial u_\tau}{\partial n} &= -\frac{\partial p}{\partial \tau} + \nu \left( \frac{\partial^2 u_\tau}{\partial \tau^2} + \frac{\partial^2 u_\tau}{\partial n^2} \right) \end{aligned}$$

It is easy to show that all the terms on the left-hand-side of the equation, as well as the tangential viscous term are identically zero. If the velocity is obtained from linear interpolation in the direction normal to the body, as is the case in the present implementation, the remaining viscous term is also zero, and the pressure satisfies  $\nabla p = 0$  on  $\Gamma$ .

A configuration close to the problem of interest, a backward-facing step, has been calculated. The results appear to be in good agreement with the experimental data. The reattachment point, for instance (Fig. 1) is predicted correctly.

Unfortunately, due to the significant amount of time spent dealing with the problems with the spectral code, the proposed cavity calculation could not be completed. The development of the present code, however, will be beneficial for future work of interest to NASA.

## References

- [1] 1999 Piomelli, U. "Large-eddy simulation: achievements and challenges." *Progress Aero. Sci.* **35**, 335.
- [2] Sarghini, F., Piomelli, U., & Balaras, E. 1999 "Scale-similar models for large-eddy simulations." *Phys. Fluids* **11**, 1607.
- [3] Spalart, P. R. & Watmuff, J. H. 1993 "Experimental and numerical study of a turbulent boundary layer with pressure gradients." *J. Fluid Mech.* **249**, 337.
- [4] Peskin, C. S. "Flow patterns around heart valves: a numerical method." *J. Comput. Phys.* **10**, 252.
- [5] Verzicco, R., Mohd-Yusof, J., Orlandi, P. and Haworth, D. "Large-eddy simulation in complex geometric configurations using boundary body forces." *AIAA J.* **38**, 427 (2000).

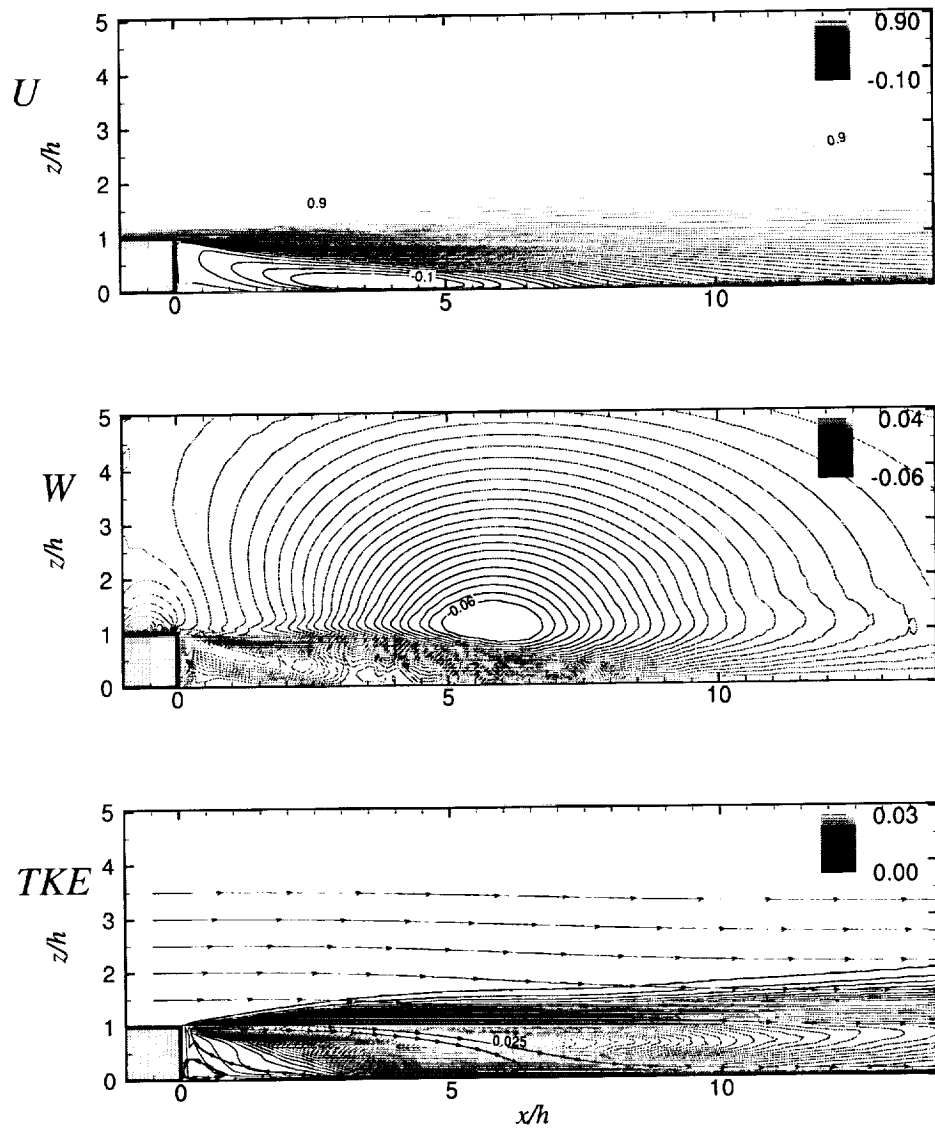


Figure 1: Velocity and turbulent kinetic energy contours in the backward-facing step.